From history

Before adopting the Gerber and ODB++ standards, no printed circuit board (PCB) manufacturing guideline existed. The materials were delivered as texts, bitmaps, and images. Data must be unified to avoid frequent misunderstandings between customers and manufacturing companies. Formats were created entirely independently of the CAE/CAD (Computer-Aided Design) software.

What data format do we work with?

1. Gerber data

Every design system can export Gerber data. This format contains the data needed to produce the PCB, i.e. conductors (paths), insulation, solder pads of various shapes, and spilt lands. Everything in this format is made up of XY coordinates, i.e. vector.

We accept **Gerber with RS-274-X** designation, which is user-friendly, especially from the point of view of import into the CAM system.

It is essential to have a **correctly configured export** from the design system, including the correct naming, which can look like this:

***.GTP (Gerber Top Paste)**

***.GTO (Gerber Top Overlay)**

***.GTS (Gerber Top Solder) – alternatively *.SMT**

***.GTL (Gerber Top Layer) – alternatively *.TOP**

***.L2 (Inner Layer 2) – alternatively *.G1**

***.L3 (Inner Layer 3) – alternatively *.G2**

***.GBL (Gerber Bottom Layer) – alternatively *.BOT**

***.GBS (Gerber Bottom Solder) – alternatively *.SMB**

***.GBO (Gerber Bottom Overlay)**

***.GBP (Gerber Bottom Paste)**

***.GM1 (outline DPS)**

The exported file can have the extension *.gbr or *.ger, and the description of the layer is given directly in the file name (TopSilk.gbr). Of course, it is possible to change the name of the target document, but we recommend keeping the one offered by the design system.

It is ideal for compressing all documents into a single zipped file and sending it to us via the web form.

If you **do not require** the application of one of the exported layers (most often, we are talking about printing), then this layer does not have to be supplied, or it states **verbatim in the specification that it should not be used**.

Descriptions of individual layers:

Silk (print)– primarily contains texts (references) to individual components, outlines and locations of electronic component cases installed from the top; often, the company logo, date code or DPS name is indicated here
Mask (non-soldering mask) – in this layer, some areas define where the non-soldering mask will not be applied and developed as a result. They, therefore, most often overlap the surfaces where the electronic components are installed
Paste (paste layer)– in this layer, some pads are 1:1 identical to solder pads for SMT assembly. The paste layer is used to create a screen printing template or to create a program for a paste-applying machine (in our case JetPrinter)
Copper Layer (TOP/BOT copper layer) - contains the central motif of the PCB, i.e. connections, insulation, and flats. After the production, this layer is copper, covered with a non-soldering mask, and surface treatment is applied to the unmasked apartments to prevent copper oxidation.
Plated Through Holes (PTH)- a coordinate file that defines the location and diameters of the holes that pass through the entire PCB.
Non-Plated Through Holes (NPTH)- a coordinate file defining the location and diameter of holes that also pass through the entire PCB. However, they are drilled in the final stages of PCB production. They are, therefore, not metallized and primarily serve as mounting holes for attachment to the last box or moulding.
Blind via (blind holes)– these holes do not pass through the entire board but connect the top layer with one of the inner layers.
Buried via (buried holes)– these are holes that connect only the inner layers and are laminated in the board
Board Dimension (board outline) - defines the physical shape and dimension of the PCB. According to this layer, the PCB is machined by milling or V-grooving.

There may be other layers, but they are no longer a common standard, for example, a layer of sensor varnish or carbon paste.

2. The ODB++ standard

The ODB++ format contains significantly more information than Gerber. Nowadays, we encounter it quite often due to its simplicity and straightforward interpretation. However, for problem-free use, it is essential to have a correctly set export already from the design system, where there are many more setting options than with classic Gerber.

In the case of a high-quality configuration, we can export, among other things, **paste data, assembly drawings and a Pick&Place file**from these documents.
In addition to the parts list (BOM), it contains all important assembly parameters. Furthermore, it will significantly speed up the process of processing the program for printing paste and the schedule for optical inspection of 3D AOI.

When using the ODB++ standard, the impact of human error on the customer's project and possible misinterpretation of data is reduced.

3. BRD file

These are data **from the EAGLE design system**. We are gradually moving away from this format because there are several serious risks when exporting to Gerber (ODB++ format is not supported). Here are some of them listed:

- Incompatibility of individual versions of Eagle
- The customer uses layers other than those taken as standard
- The font used can be defined as either vector or proportional at the input, which may cause a change in the length of the text during export

After prior agreement, however, we can process this format, we have to agree on the production data before starting production, or we can agree on customer export using a so-called CAM processor.

We consider the following to be inappropriate data (from our point of view unprocessable):
dxf, dwg, step, diagram (in any form)

The customer bears full responsibility for the correctness of the documentation.

In conclusion

From a PCB designer's point of view, it is a good idea to have the exported files tested and reviewed in the CAM viewer just before sending them to the manufacturer.
At Safiral, we use several CAM systems to read data, depending on whether the sales department, TPV or AOI control department is working with the data. We work with PCB designers who have experience with the specifics of our production and are regularly introduced to our technological innovations and requirements for the supplied materials.
Would you like to get in touch with certified PCB designers? Contact our sales department. We will be happy to connect you with our partners.